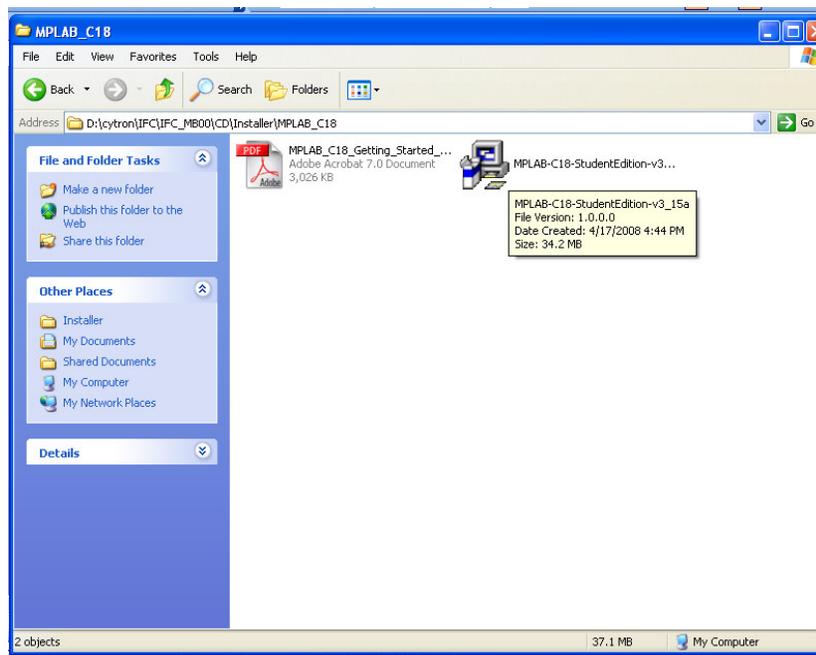


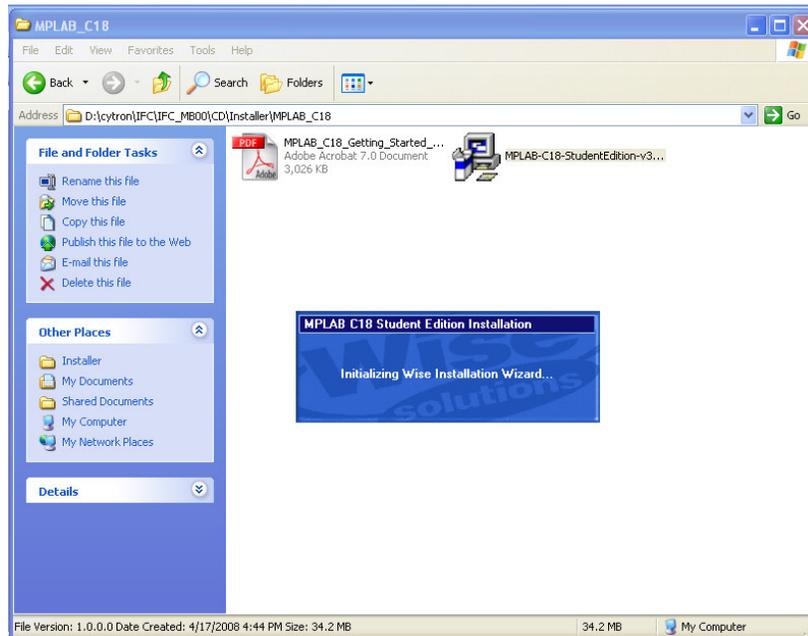
## MPLAB C18 Compiler Installation

The MPLAB<sup>®</sup> C18 compiler is a full-featured ANSI compliant C compiler for the PIC18 family of PICmicro<sup>®</sup> 8-bit MCUs. Microchip offer free MPLAB C18 compiler for student with no limitation of code size and PIC type; the only drawback is the code optimization is disabled. For details description of MPLAB C18, please refer to MPLAB C18 User's Manual. Following diagrams show the installation guide of Student Version of MPLAB C18 compiler. Please install MPLAB C18 according to following step.

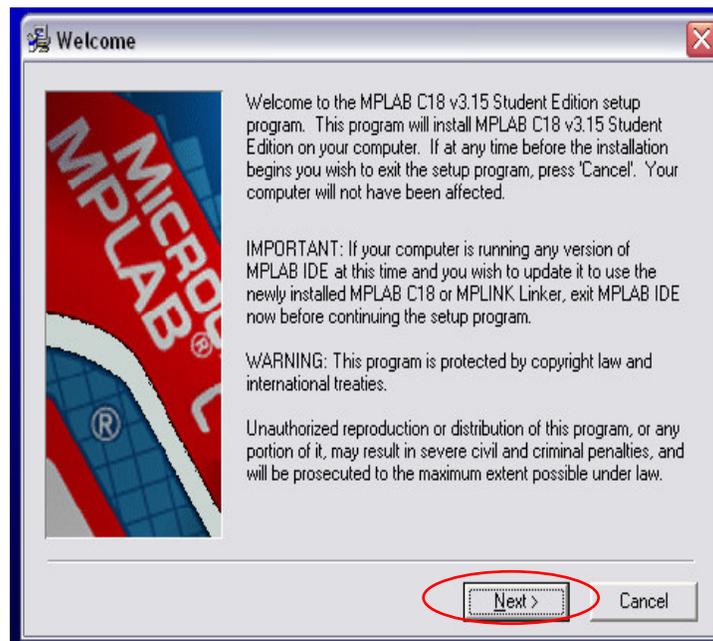
1. Please double click on the application file for MPLAB-C18-StudentEdition-v3\_15a to start the installation.



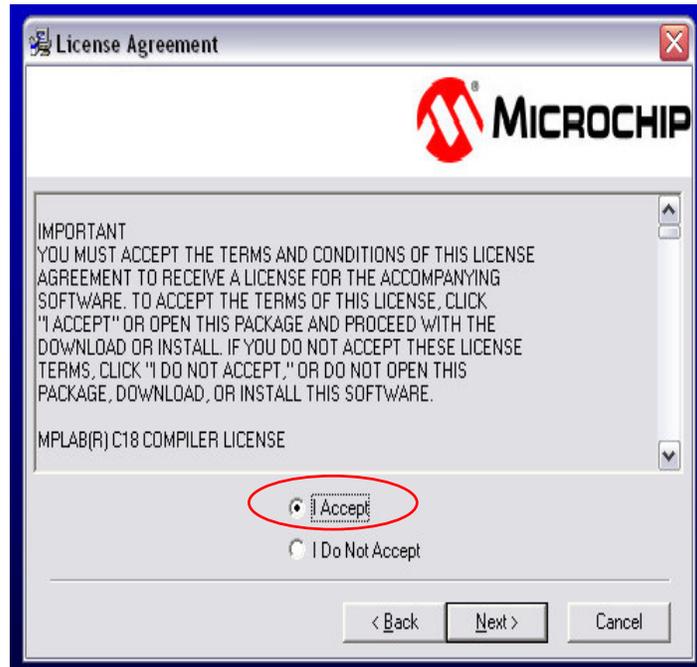
2. An Initializing Wizard that looks like the diagram below will pop out



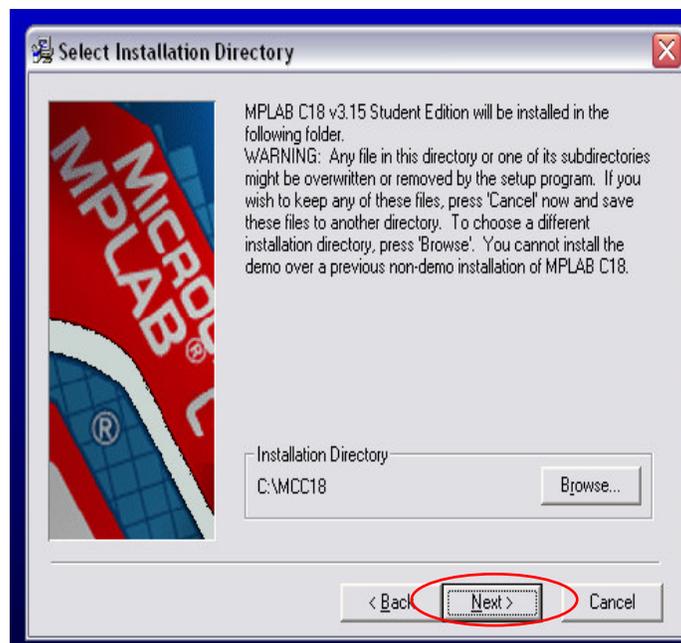
3. After that, there will be a welcome screen displays the version number of MPLAB C18 that the setup program will install. Please click *Next>* to proceed.



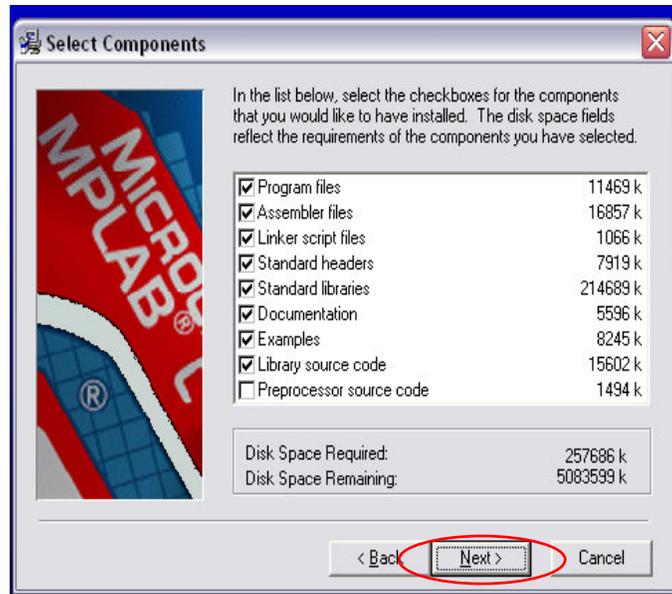
4. For License Agreement, please select *I Accept* if it is not selected in the diagram below, and click *Next>* after this.



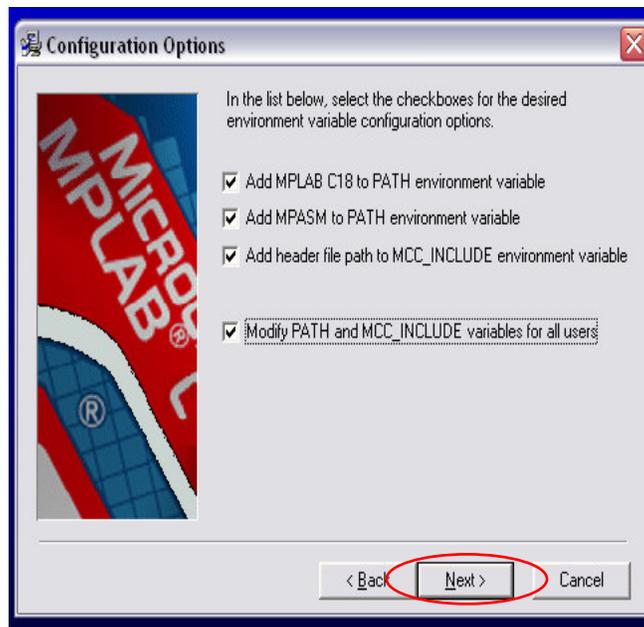
5. In the *Select Installation Directory*, user is recommended to use the default installation directory, which is *C:\MCC18* or user is also free to select the other installation directory by using the *Browse...* button to browse to the desired destination folder. After that, click *Next>* for next step.



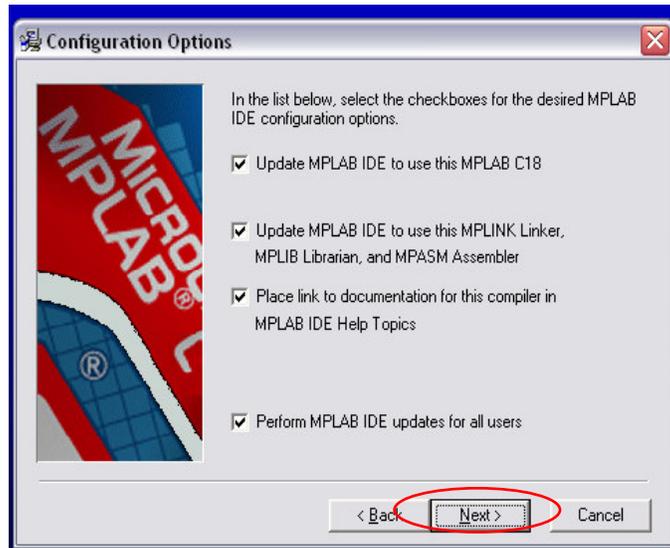
6. User can choose the components to be installed by checking the appropriate box in *Select Components* window. Please refer to MPLAB-C18-Getting-Started document in the CD for detailed description of the available components.



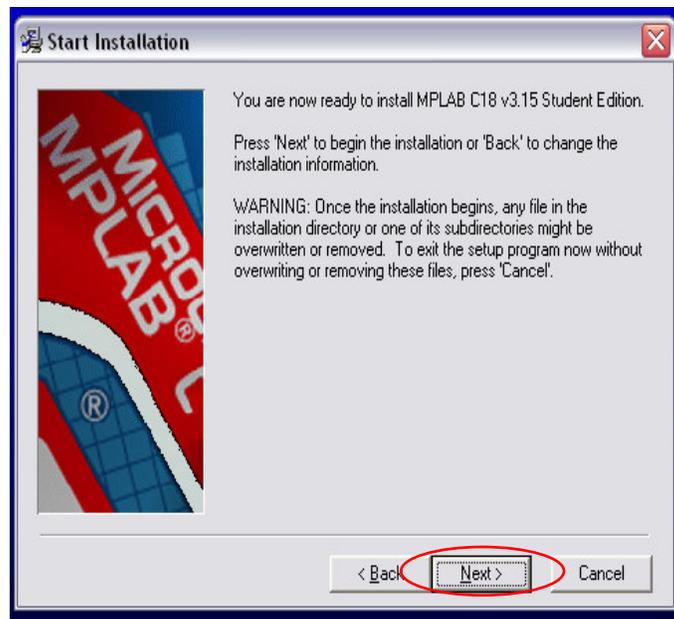
7. The *Configuration Options* below allows users to select a particular set of desired environment variable configuration options for their system. Please refer to MPLAB-C18-Getting-Started document in the CD for detailed description of the available configuration options. In this example, all the options are selected. Please click *Next>* to continue.



8. There is another *Configuration Options* which allows users to select a particular set of desired MPLAB IDE configuration options for their system. Please refer to MPLAB-C18-Getting-Started document in the CD for detailed description of the available configuration options. In this example, all the options are selected. Please click *Next>* to continue.



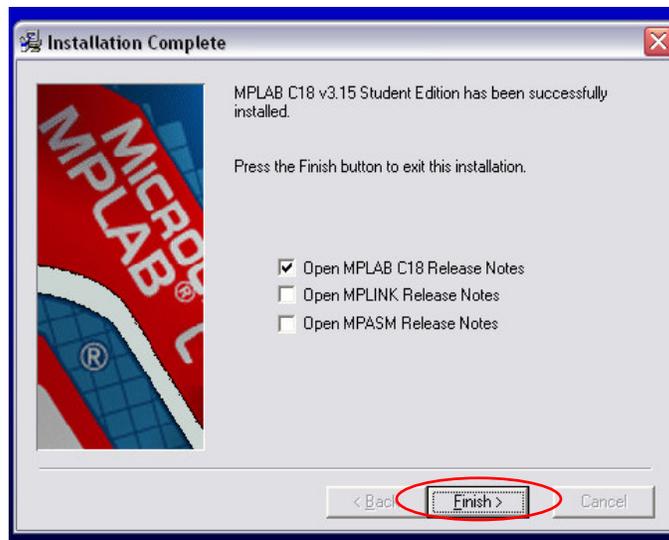
9. Please click *Next>* for the diagram below to start installation after all the setting done.



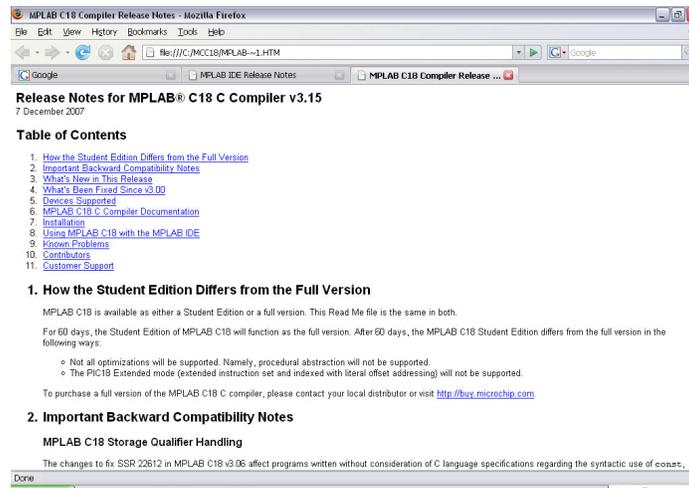
10. Please wait for awhile for the installation process shown in diagram below.



11. After the installation completed, a diagram as below will be shown, user can select the release notes that wish to read, in this example, the release notes for MPLAB C18 is being selected. Please click *Finish*> to exit installation.



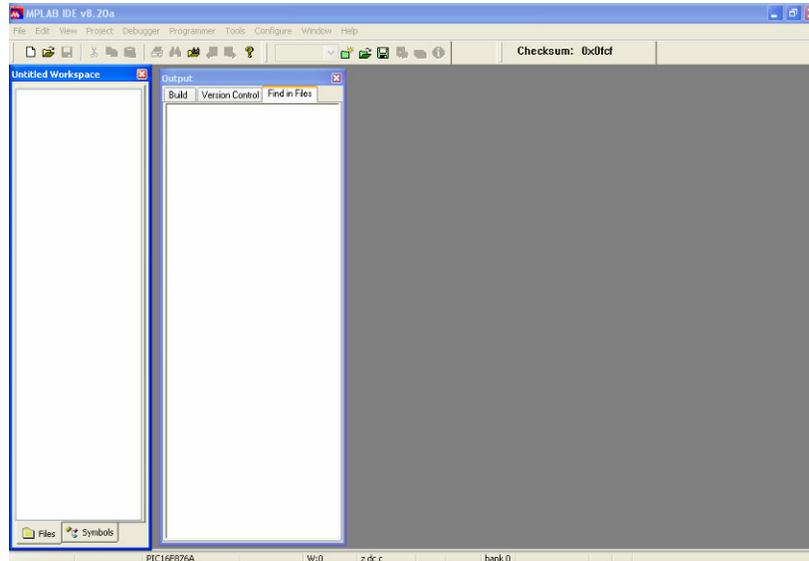
12. The release notes MPLAB C18 is being shown after the installation process since it was being selected in the last step.



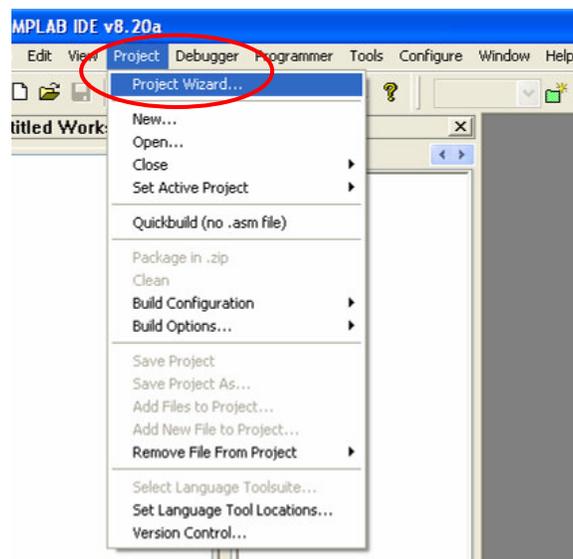
## Open Project for PIC18F series.

To start MPLAB IDE and open project for PIC18F series, please follow the step below:

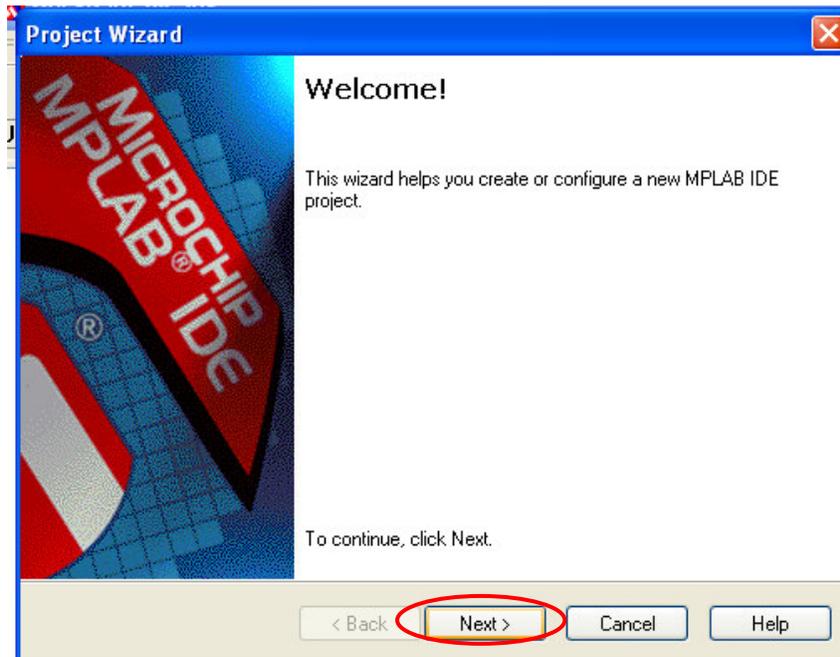
1. Double click on the icon installed on the desktop after installation or select *Start>Programs>Microchip> MPLAB IDE v8.20a>MPLAB IDE*. A screen will display the MPLAB IDE logo followed by the MPLAB IDE desktop as in diagram below.



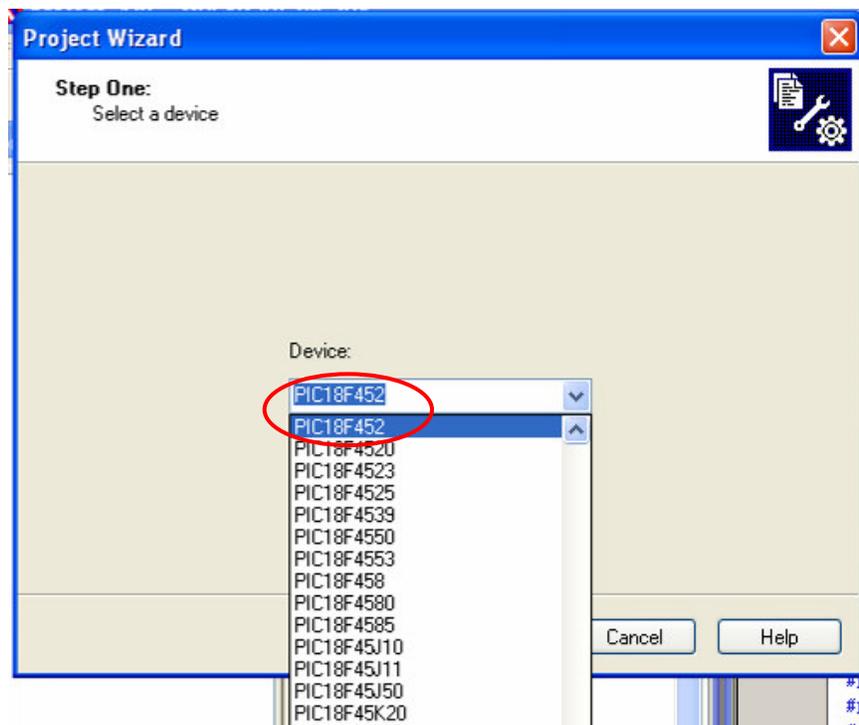
2. The next step is to create a project using the Project Wizard. A project is the way the files are organized to be compiled and assembled. We Choose *Project>Project Wizard*.



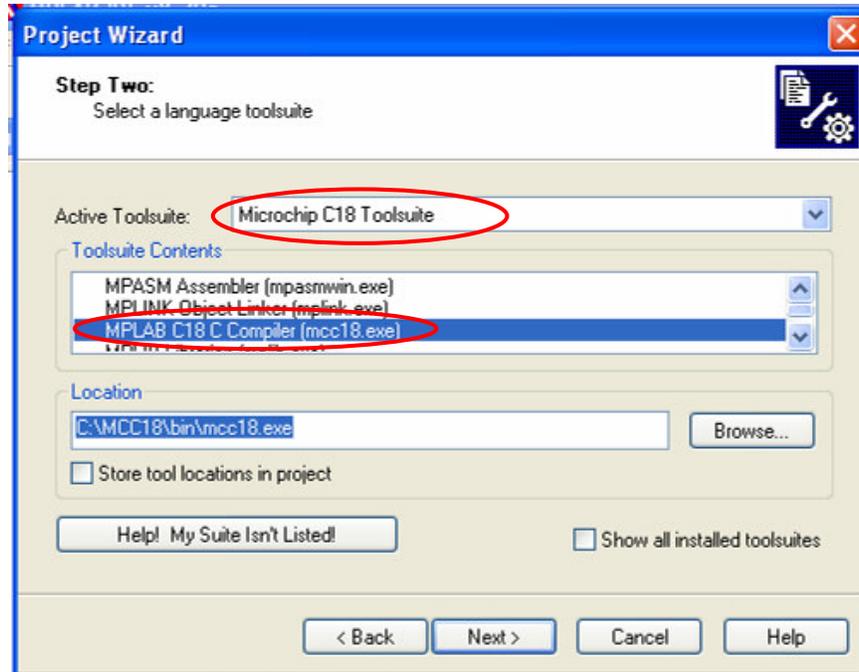
3. From the Welcome dialog, click on *Next* to proceed.



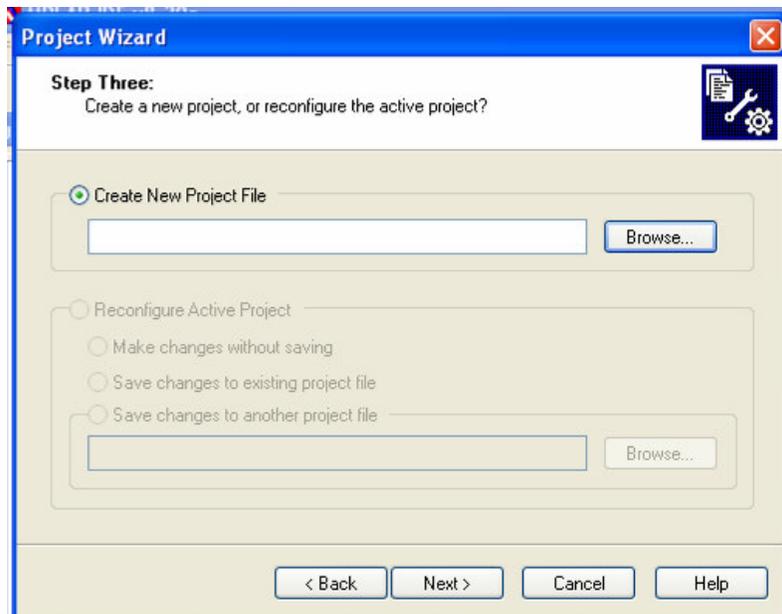
4. The next dialog (Step One) allows you to select the device. In this example, PIC18F452 was selected from the drop down menu. Click *Next*>.



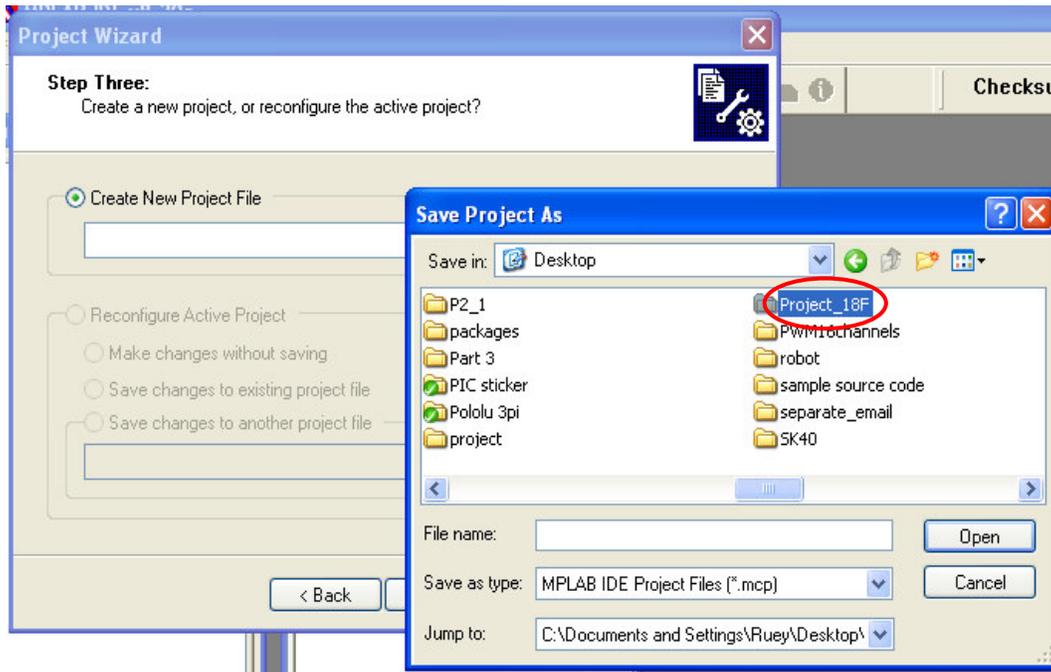
5. The next step of the Project Wizard is sets up the language tools that are used with this project. Select “Microchip C18 Toolsuite” in the Active Toolsuite list box. Then select “MPLAB C18 C Compiler (mcc18.exe)” in the Toolsuite Contents box. When you are finished, click *Next*.



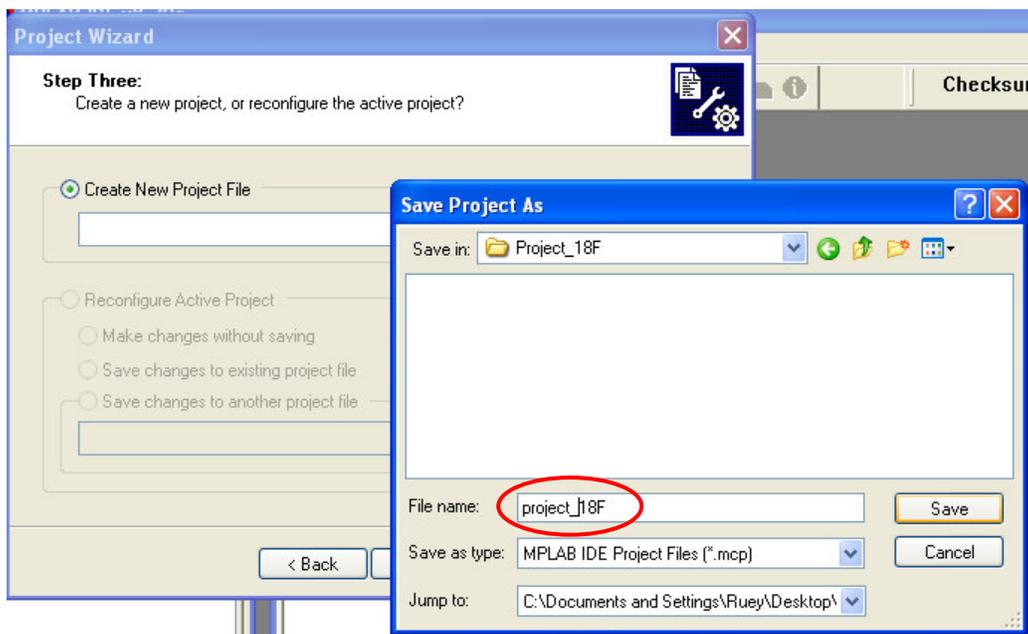
6. Step three of the project wizard allows user to create new project file.



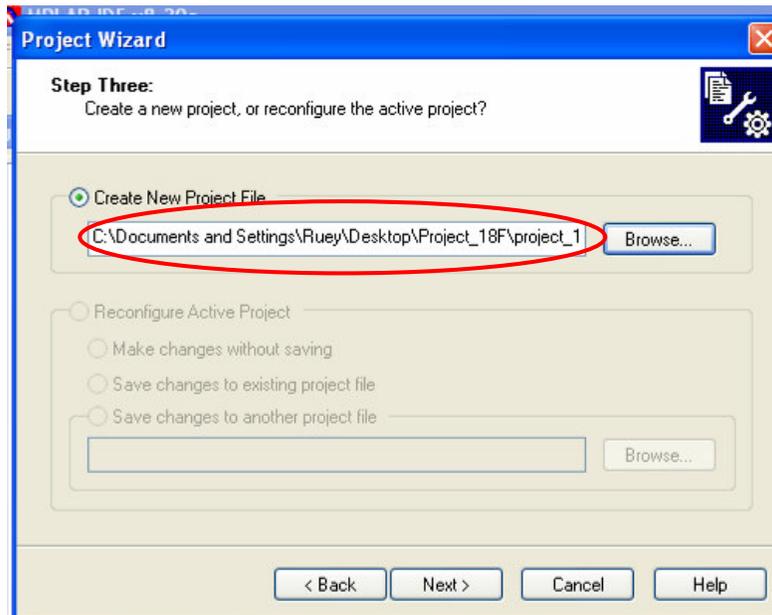
- For an example, a folder named *Project\_18F* was first created at Desktop.



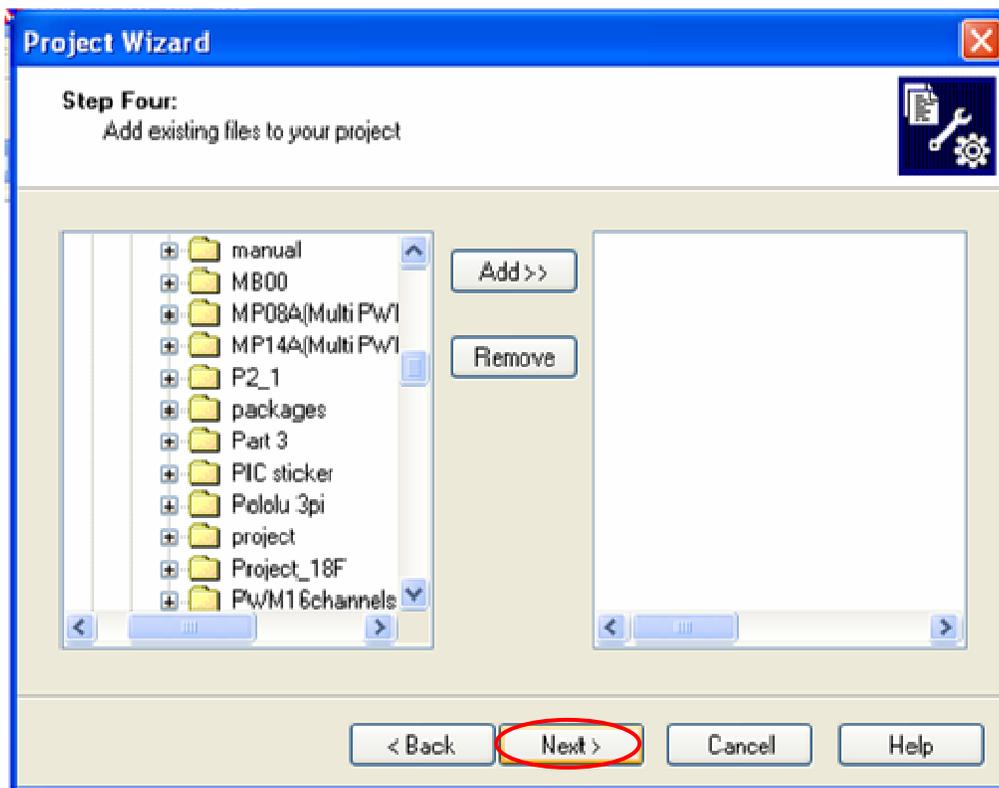
- Then open the folder, *Project\_18F*. Project named “*project\_18F*” can be created by typing the project name in the column for “File name”, and click *Save*.



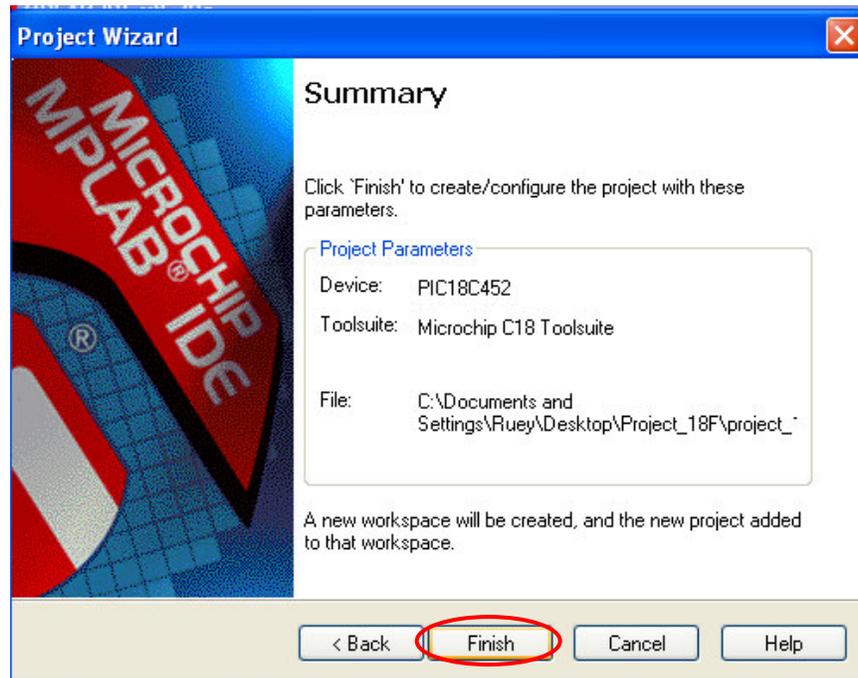
9. Diagram below shown the Project “project” had been created and the directory. Click *Next*>.



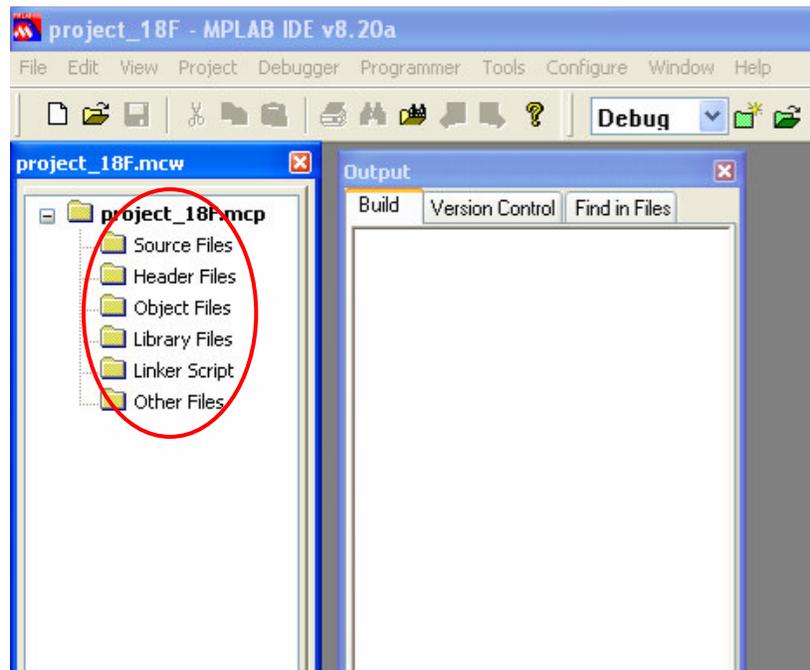
10. Step four of the project wizard allow user to add existing file to the project, however, for this example, no files will be added. Please click *Next*> to proceed.



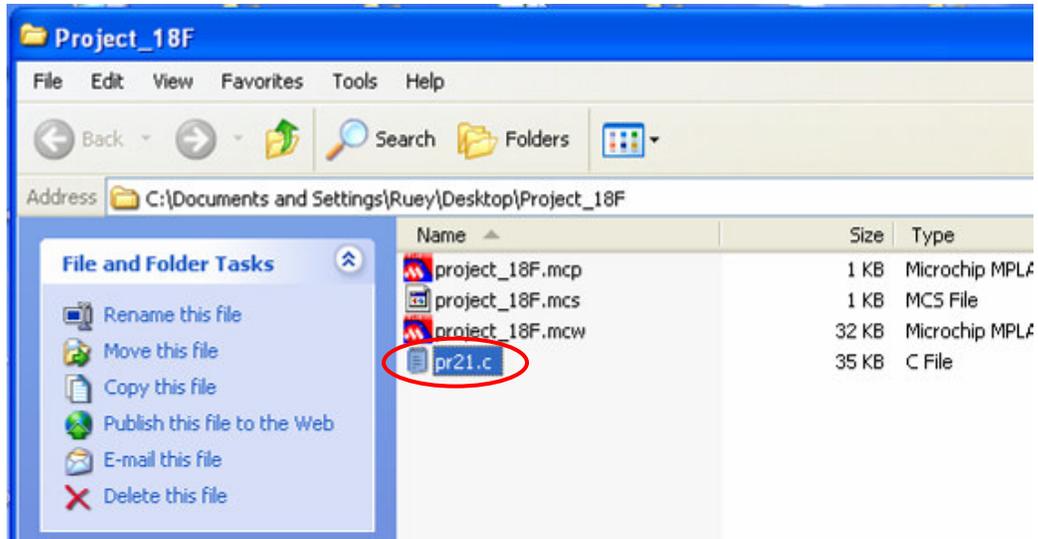
11. A summary will be shown at the end of project wizard, all the project parameters are shown. Please click *Finish* to exit from project wizard.



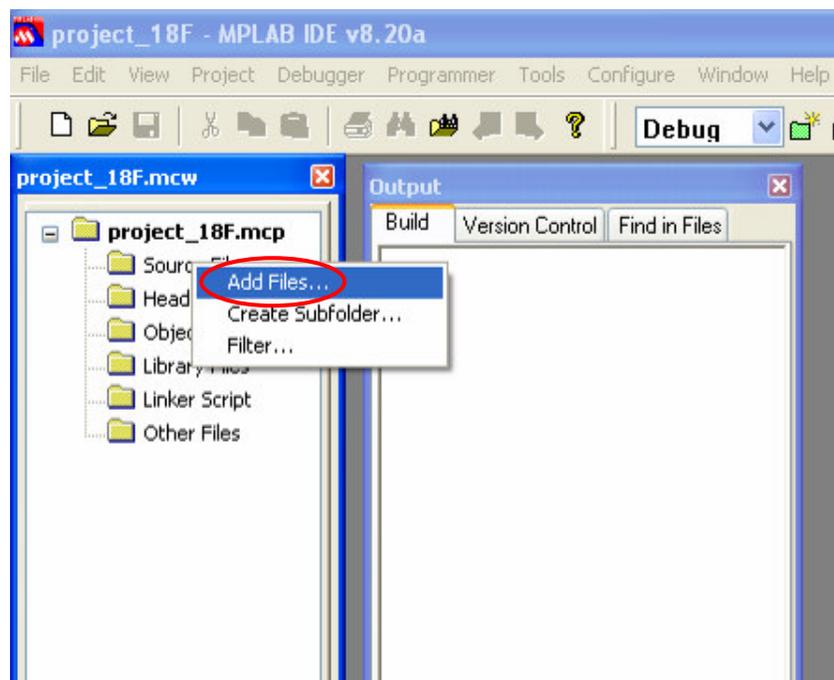
12. After pressing the *Finish* button, review the Project Window on the MPLAB IDE desktop. It should look like the diagram below. If the Project Window is not open, please select *View>Project*.



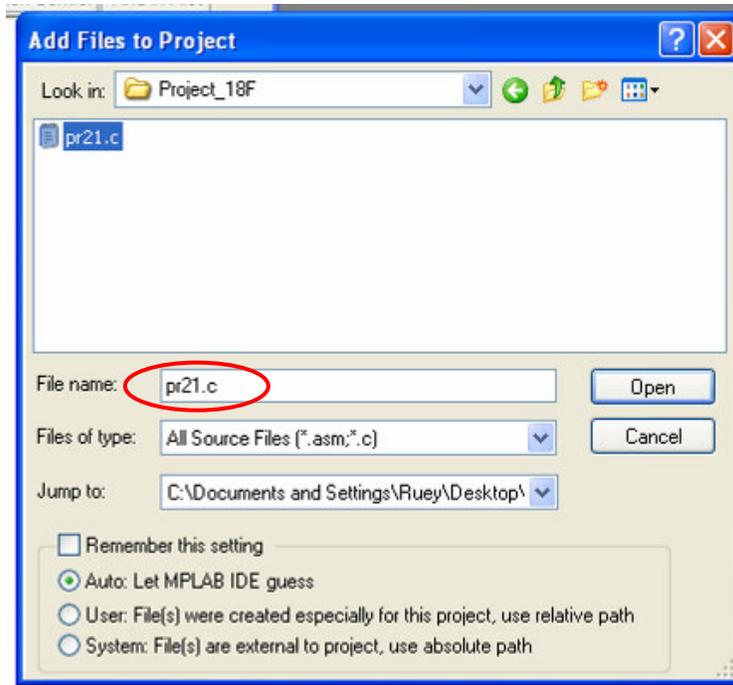
13. In this example, sample source code for Cytron DIY project, PR21 will be added to this project. The sample source code can be downloaded at <http://www.cytron.com.my/PR21.asp> . Diagram below show the sample source code, pr21.c being copied and pasted in the folder, *project*.



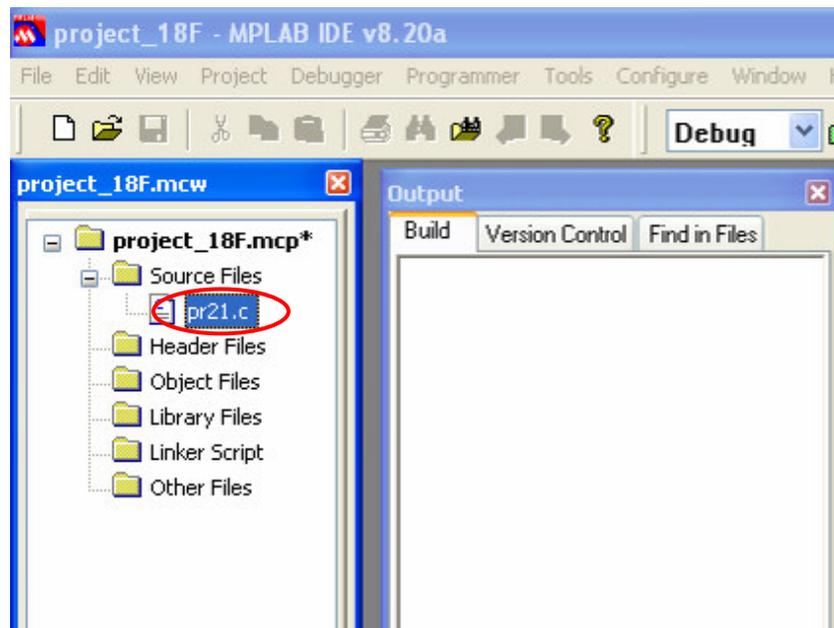
14. To add file in *Source Files*, right click on the *Source Files*, then click on *Add Files...*, diagram below shown the example for add file to *Source Files*



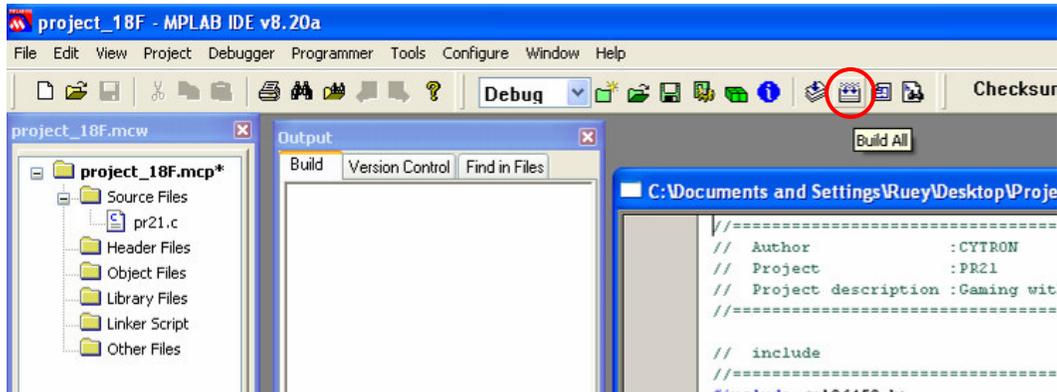
15. After clicking on *Add Files...*, a window pop out, do make sure the *Files of type* is *All Source Files (\*.asm;\*.c)*, then browse to the folder *Project* to add in “pr21.c”. User can select the file, “pr21.c”, and click open to add the file.



16. Diagram below shown pr21.c added to the project.



17. After added the source file, user can open pr21.c file in this workspace and try to compile it. Diagram below shown opened pr21.c file. To compile, user can go *Project>Build All* or the build icon (in red circle) on menu bar as shown in diagram below.



18. After build success, a message *BUILD SUCCEEDED* will appear in output window like shown in diagram below.

